Fusion3

F200 3D Printer

RUNNING A PRINT

Revision 7/17/2025

RUNNING A PRINT (AKA NORMAL OPERATION)

The steps, procedures, & what to expect when printing with the F200.

SUMMARY

There are 11 key steps to run a print on Fusion3 printers:

1. Slicing your part in F3Slic3r
2. Transferring your .gcode file to to the printer
3. Loading filament
4. Preparing the bed
5. Starting the print
6. Start-of-print sequence
7. Ensuring a correct first layer
8. The printing process
9. End-of-print sequence
10. Removing your finished part from the printer
11. End-of-print cleanup

STEP 1: SLICING YOUR PART IN F3SLIC3R

The basic steps are:

1. Export your CAD file as a .STP or .STL file. .STP is preferred for best print quality.
2. Load this into F3Slic3r
3. Configure your options & settings in F3Slic3r (material, quality preset, etc)
4. Generate the .gcode
5. Preview / check the .gcode
6. Save the .gcode file

For more information see:

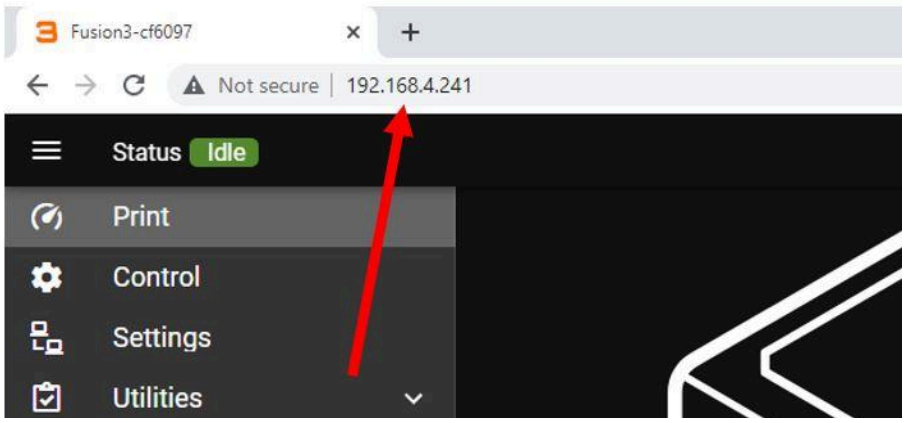
- Our F3Slic3r training video: https://youtu.be/w3l7Vgb0iKc

STEP 2: TRANSFER YOUR .GCODE FILE TO F200

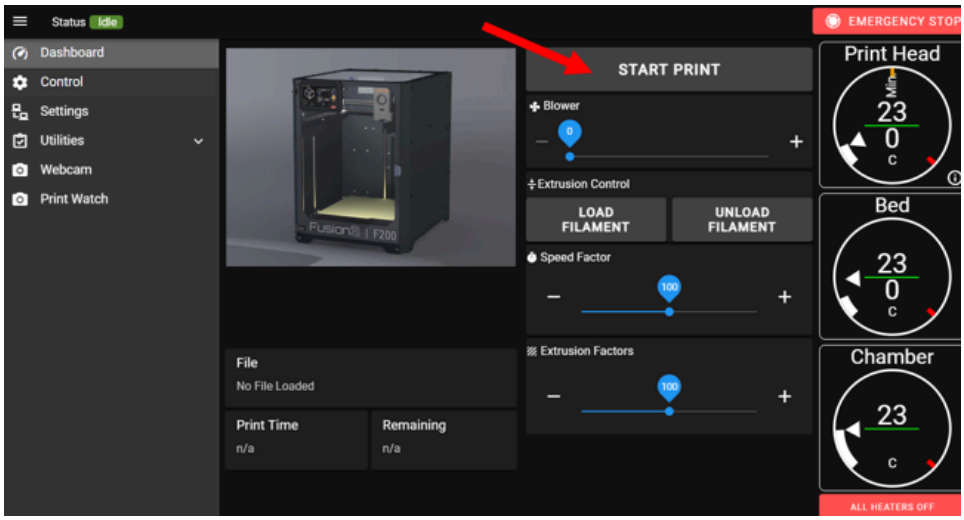
There are two ways to transfer your .gcode file to your printer:

1) If Your Printer Is On Your Local Network

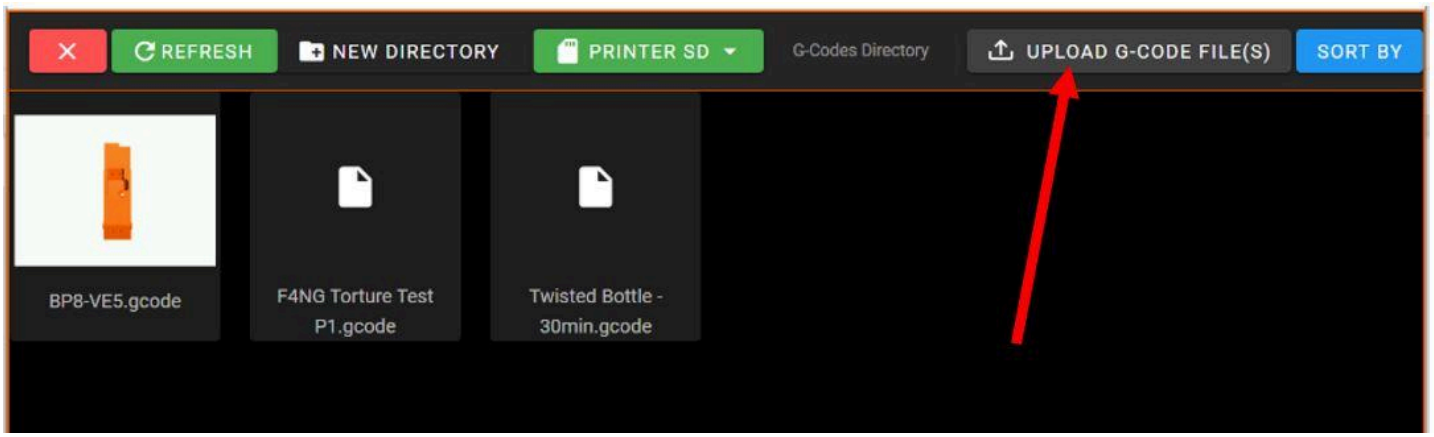
1. Load your printer's user interface on your computer by navigating to its IP address with a web browser.



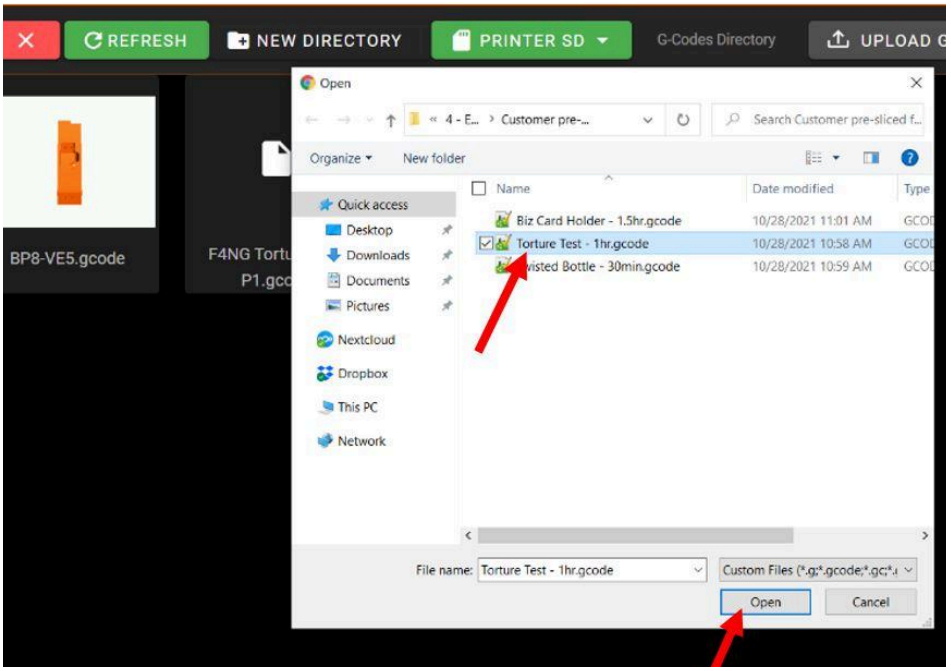
2. Click the "START PRINT" button on the main screen.



3. In the file selection dialog box, click "UPLOAD PRINT" in the top right corner



4. In the dialog window that opens, navigate to where your .gcode file is saved on your computer. Select it and click "Open".
5. The file will upload and appear in the file list.
6. **DO NOT** select it for printing yet; we have to prepare the printer itself!



2) Manual Transfer Via USB

If your printer is not on your network, or you do not wish to use the network interface:

1. Locate a USB thumb drive and insert it into your computer. Transfer your .gcode file to it.
2. SAFELY EJECT the thumb drive and transfer it to the USB port on your printer's back panel
3. **THAT'S IT**, until we've prepared the printer for printing.

STEP 3: LOADING FILAMENT

See the "*Loading Filament*" video HERE for detailed information on how to load filament into your printer.

Ensure that the material you load MATCHES the .gcode file you made! For example, don't load PLA if you have sliced for ABS.

STEP 4: PREPARE THE PRINT BED

The PEI print surface requires less prep than a glass print surface since no glue is required, but the surface does need to be cleaned every so often, especially if you encounter first layer adhesion issues. Our cleaning and use recommendations for the PEI surfaces are HERE.

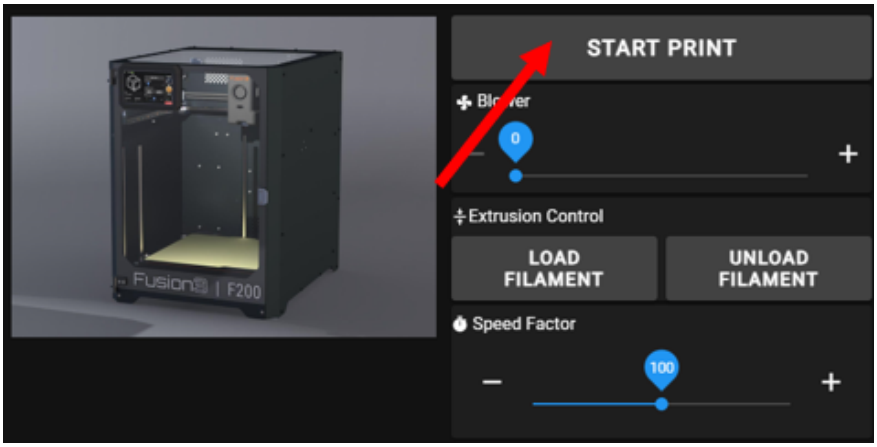
Ensure that the bed you have installed on your printer and its preparation matches the requirements of the material you're using, **and** your slicer settings.

Ensure the bed is installed correctly and is not sitting on top of any of the alignment guides. See HERE for more info.

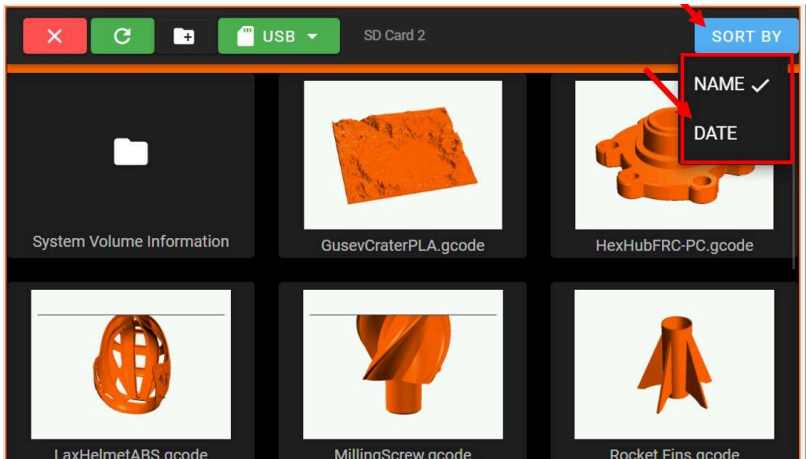
For the PEI sheets, there's no need to heat soak the print surface. By the time the printer goes through its nozzle offset and bed leveling process, the bed should have had plenty of time to come up to temperature on the actual print surface.

STEP 5: START THE PRINT

1. On the main screen, press the "START PRINT" button.

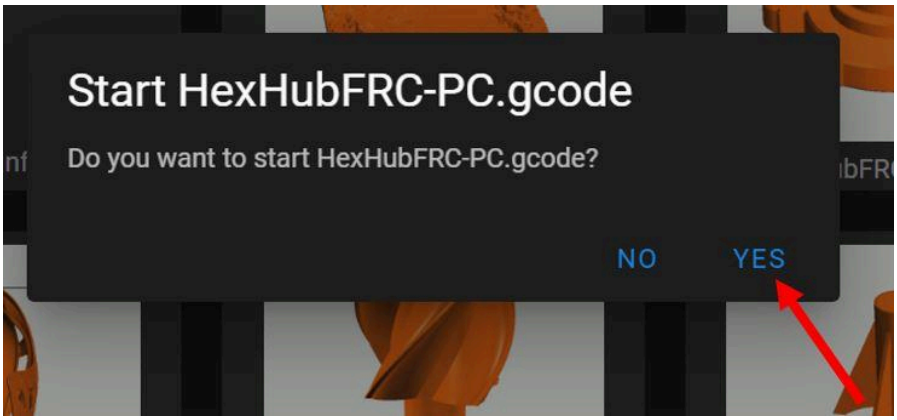


2. A file selection dialog window will pop up. Scroll through the file list to find the file you want to print.
    a. Thumbnail images will show what the prints look like
    b. Use the "SORT-BY" button in the top right to sort the files by name, or by date created. The latter will put your most recent file first which makes it easy to find.



c. A long press on a file will bring up additional options.

3. Select the file you want to print. Press "YES" to confirm you want to start the print immediately.



4. The printer will begin executing the print.

STEP 6: BEGINNING-OF-PRINT SEQUENCE

At the start of each print, the printer will complete the following steps:

1. **Bed Heating**: The bed heats to its target temperature. Nothing else will happen until the bed is up to temperature. You will see the bed temperature gauge pulse orange to indicate it is heating, and the UI will transition to its print progress screen. For ABS or other high temperature materials, heating the bed for the first time may take up to 20 minutes.
2. **Home Axes**: Once the bed is up to temperature, the printer will home its Z, Y, and X axes, in that order.
3. **Bed Leveling, Nozzle Offset, Heat Print Head**: Your printer will perform a calibration routine that cleans the print head, checks the nozzle offset, and checks the bed level. The print head will heat up during this sequence. Watch the screen for status updates and information on what the printer is doing.
4. **Prime Print Head**: Once the bed probing is complete, the print head will move to the front left corner, extrude some filament in a line on the print bed, then the print will begin.

NOTE: The "pause print" functionality is not available until your printer actually begins to print your part. If you need to stop your print during this time, press the E-STOP button in the top right corner of the LCD, or power cycle the printer.

NOTE: If you have enabled quick-turn mode, the start-of-print sequence bypasses the bed leveling and nozzle offset checks, to speed up how quickly the print starts. Instead, the axes will home, a single probe is done in the center of the bed, then the print head heats up to begin the print.

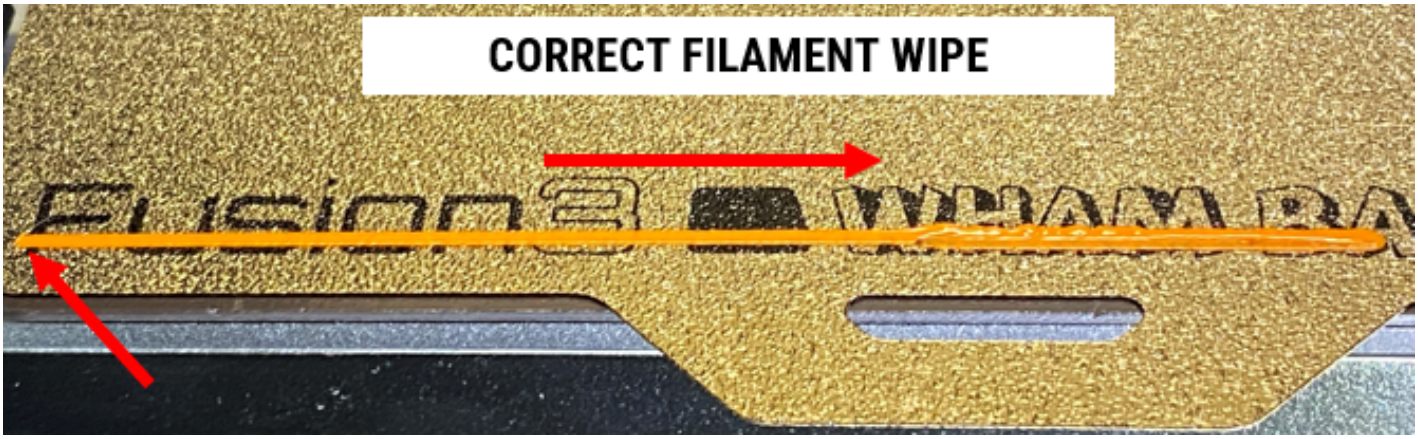
STEP 7: DURING THE FIRST LAYER

IMPORTANT: We STRONGLY recommend you ALWAYS monitor the start-of-print sequence and the printing of the first layer. Most print failures result from problems on the first layer that are not caught and corrected promptly.

To ensure the success of your print you should evaluate the following during the first layer of the print:

Filament Prime & Wipe:

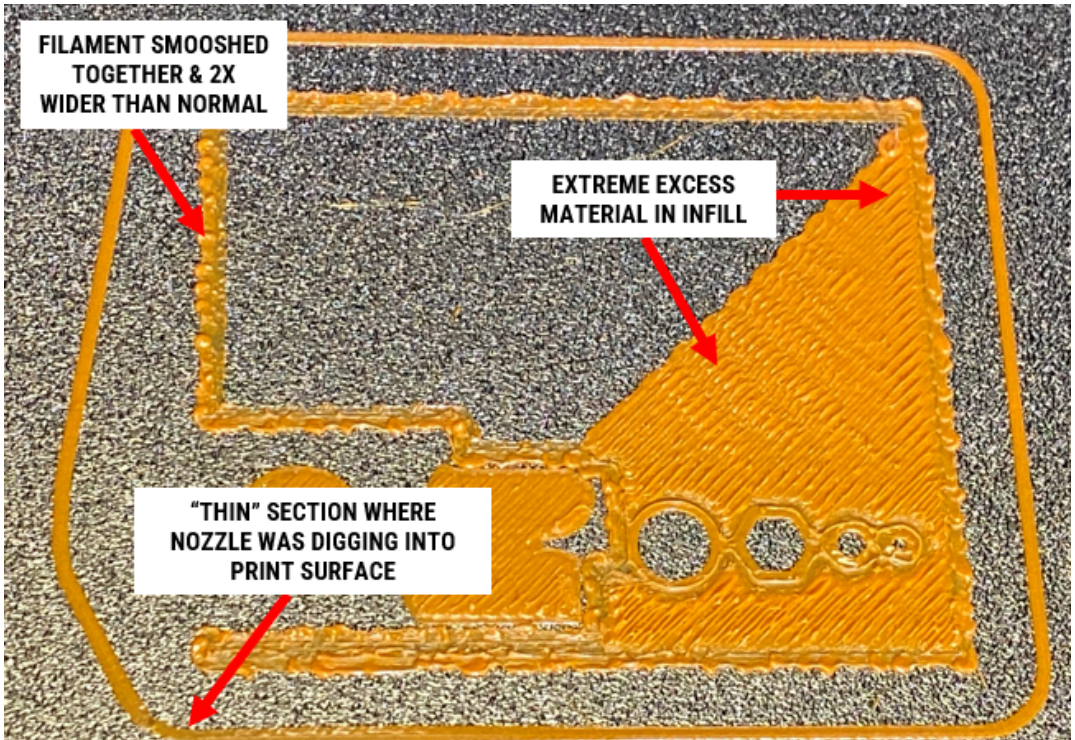
The printer will extrude a length of filament at the front left corner of the bed, then move down and "wipe" the print head left to right. Sometimes this filament will get stuck on the print head and get dragged into the print. **If this happens, you should pause the print and remove it from the print head using needle nose pliers or similar.**



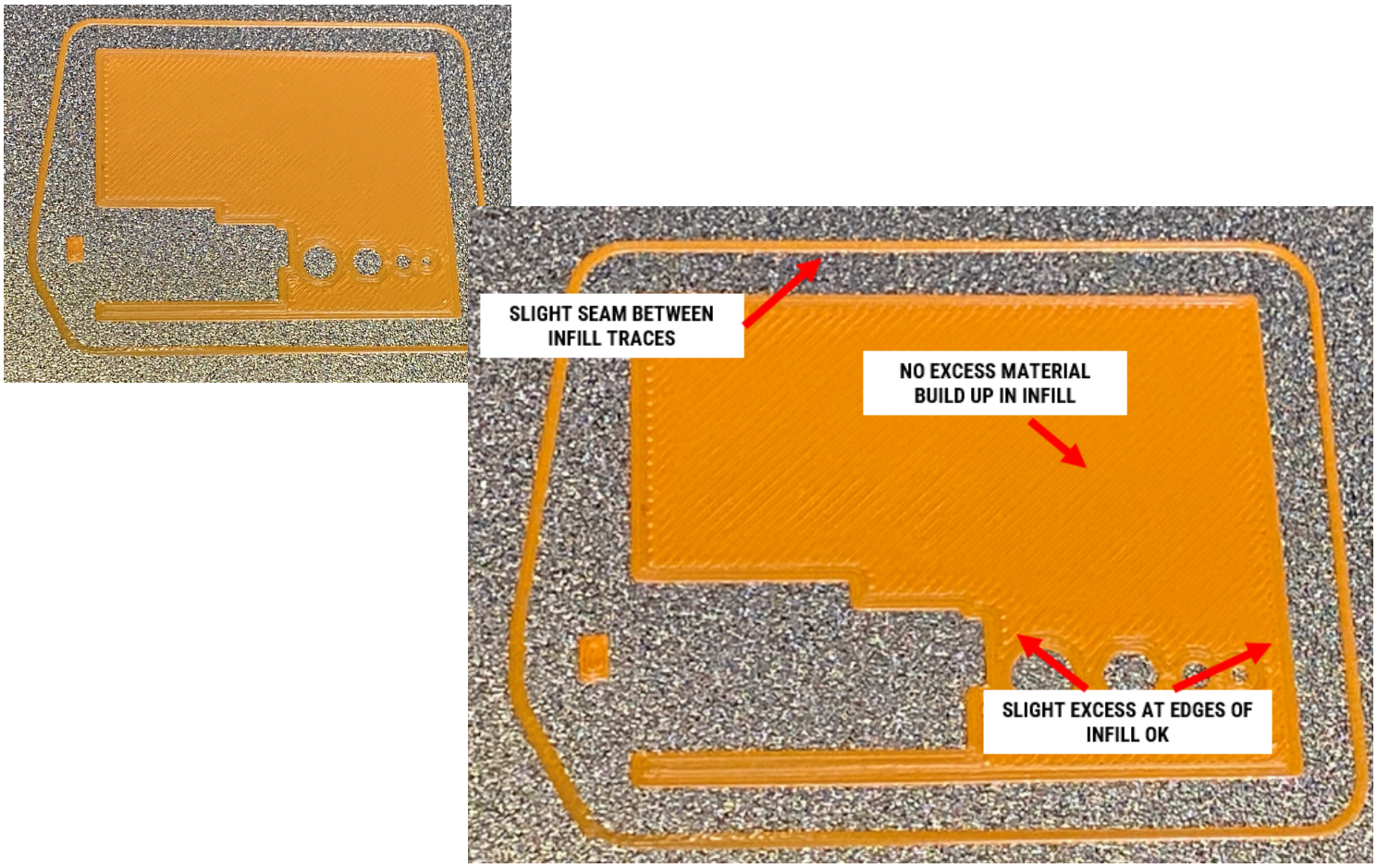
First Layer Height:

Check that the first layer height is correct. You can do this by inspecting the perimeter "skirt" lines the printer will make around the outline of your part. See pic for what they should look like.

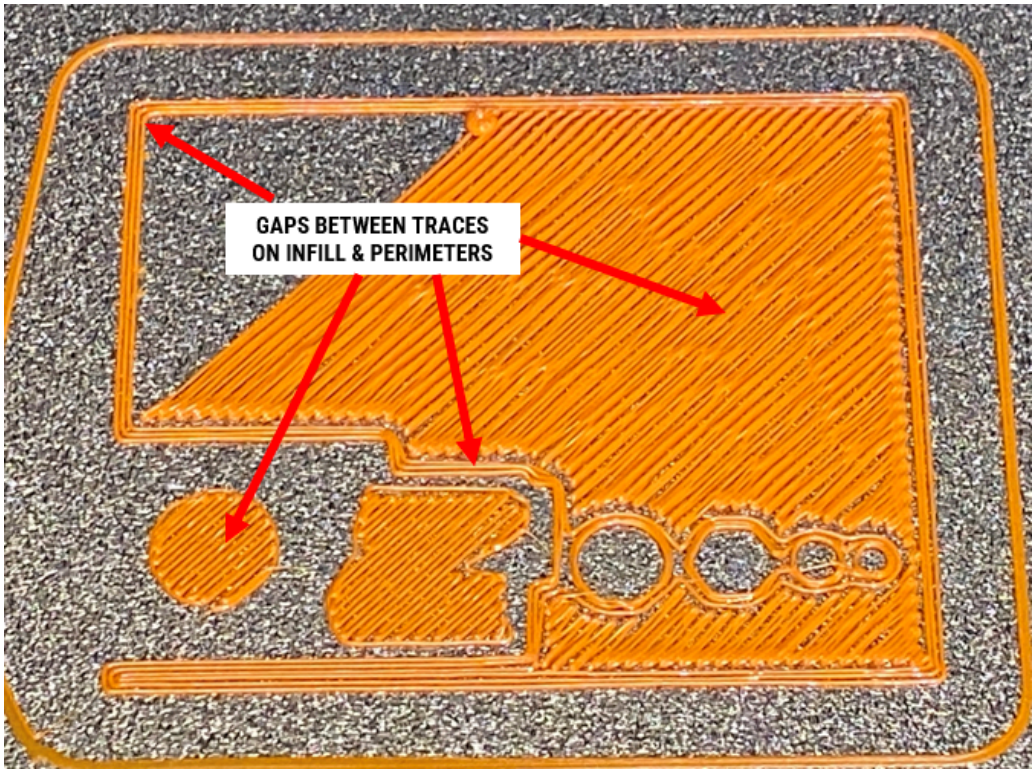
First layer FAR TOO close. STOP PRINT, fix, try again:


FILAMENT SMOOSHED TOGETHER & 2X WIDER THAN NORMAL
EXTREME EXCESS MATERIAL IN INFILL
"THIN" SECTION WHERE NOZZLE WAS DIGGING INTO PRINT SURFACE

First layer "goldilocks" - Just Right:



First layer TOO FAR AWAY. Stop print, fix, try again:



If you're not sure about your first layer height from the skirt lines, you can also look at the infill pattern. See the pictures above for examples.

If your first layer height is not correct, use the Babystepping buttons to adjust it. For more information on Babystepping see the *Bed Leveling* module HERE

Bed Adhesion:

Make sure the entire first layer sticks firmly to the bed. If you see a corner or other section lift or peel off, **stop the print**. Check the following:

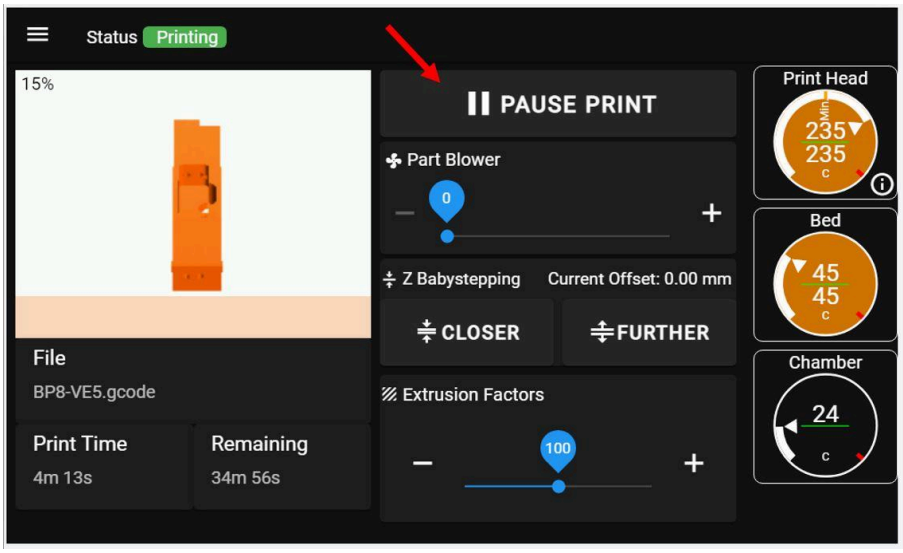
- Your slicer settings & material loaded match
- Make sure to clean the PEI sheet with dish soap and water.
- Your bed **coating** is correct for the material AND your print surface (gluestick, magigoo, nothing, etc)

STEP 8: DURING A PRINT

NOTE: If you open the main door during a print, the printer will automatically pause the print for safety reasons. To resume a print, close the main door and select "Resume" on the control panel.

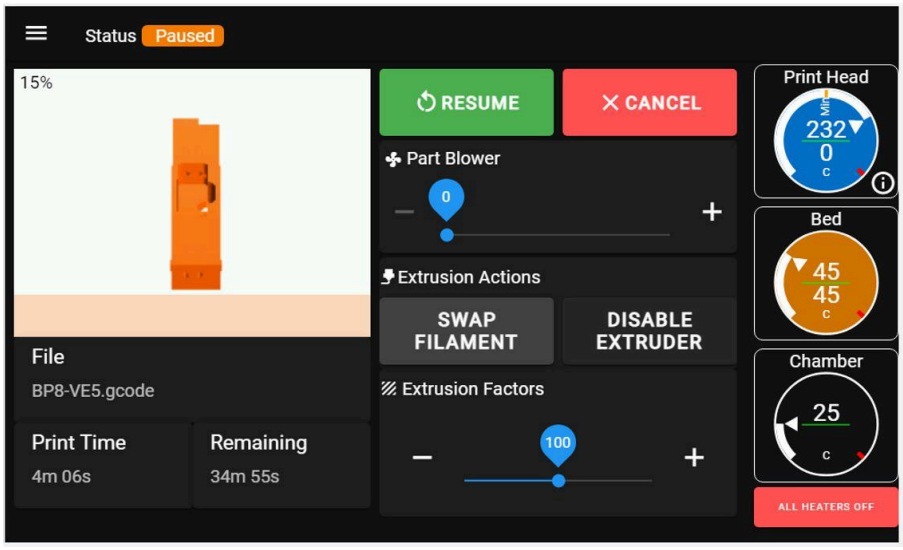
Pausing, Resuming, Canceling A Print:

To pause a print, press "*PAUSE PRINT*" on the control panel.



To resume a print, press "*RESUME PRINT*" on the control panel.

While a print is paused, you will get additional options on the control panel to swap filament, or cancel the print.



When a print is paused:

- The print head will be turned off
- The bed will stay hot
- The motors will stay energized

When you resume a print, it will look like nothing is happening while the print head is heated up. The longer the print has been paused, the longer this will take.

STEP 9: END OF A PRINT

When the print is finished, the printer will move the head to the back-left corner, and move the bed all the way to the bottom.

- If you enable quick-turn mode, the bed will stay hot.

For high temperature materials (ABS, PC, PC-PBT, etc), you may want to leave the door closed while the bed cools down. This helps the part cool more evenly and makes it less likely to warp after it's been removed from the bed.

STEP 10: REMOVING YOUR FINISHED PRINT

NOTE: If you have enabled quick-turn mode, you will need to remove your print surface from the printer in order for it to cool down because the bed will stay hot.

For detailed information on removing parts, see the *Removing a Finished Part* module HERE

Parts printed on the standard PEI surface can be popped off by removing the plate and flexing it. For large parts in high temp materials we still recommend letting the bed cool to avoid post-print warping.

STEP 11: END-OF-PRINT CLEANUP

1. Remove the priming line and print skirt from the bed.
2. (Flexible Surface) Avoid touching the surface with your hands so you don't transfer skin oil. Wash the surface periodically with dish soap.